Model created in COMSOL Multiphysics 6.4

Thermophoresis

Introduction

In a gas at nonuniform temperature, suspended particles tend to move from regions of high temperature to low, due to the *thermophoretic force*. This effect can be used to create thermal precipitators that filter out undesirable particles from a feed gas. It can also be used in chemical vapor deposition, to inhibit the arrival of particle contaminants on the surface of a susceptor. This tutorial predicts the size of a particle-free zone above a heated susceptor for different temperature gradients. The results agree well with Ref. 1.

Note: This model requires the Particle Tracing Module, the Heat Transfer Module, and one of the following: CFD Module, Microfluidics Module, or Plasma Module.

Model Definition

Hydrogen gas is injected into the modeling domain at a flow rate of 2000 SCCM. The gas flows over a heated susceptor and out through a pump after a 90-degree bend, see Figure 1. Particles are injected into the gas stream at the beginning of the susceptor uniformly in the y direction. The initial particle velocity is set to the fluid velocity.

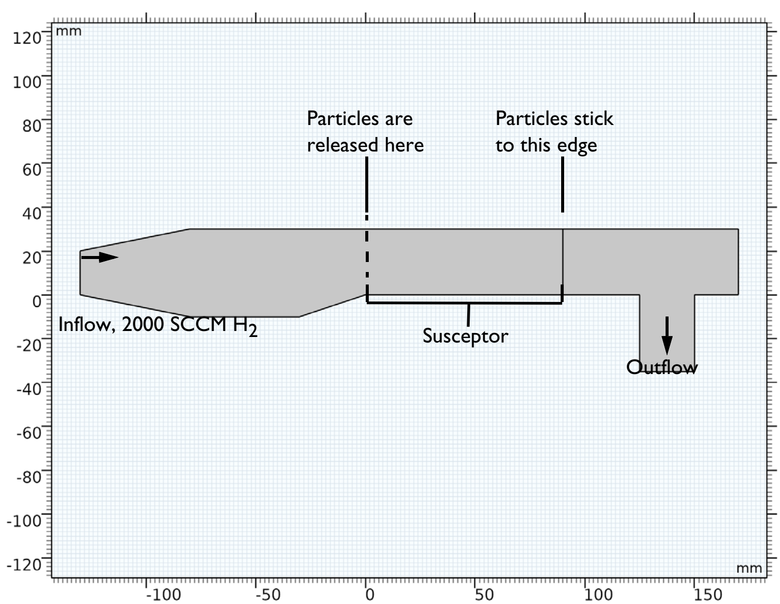

Figure 1: Diagram showing the model geometry (in mm) and location of the susceptor.

DRAG FORCE

The particle positions are computed by solving second-order equations of motion for the particle position vector components, following Newton's second law,

$$\frac{d\mathbf{q}}{dt} = \mathbf{v}$$

$$\frac{d}{dt}(m_p\mathbf{v}) = \mathbf{F}_t$$

where

- $\mathbf{q}$ is the particle position (SI unit: m),
- $\mathbf{v}$ is the particle velocity (SI unit: m/s),
- m_p is the particle mass (SI unit: kg), and
- $\mathbf{F}_t$ is the total force (SI unit: N).

In this example, the total force includes the drag, gravity, and thermophoretic forces,

$$\mathbf{F}_t = \mathbf{F}_D + \mathbf{F}_g + \mathbf{F}_{tp}$$

The drag force $\mathbf{F}_D$ is defined using the Stokes drag law,

$$\mathbf{F}_D = \left(\frac{1}{\tau_p}\right)m_p(\mathbf{u} - \mathbf{v})$$

where $\mathbf{u}$ and $\mathbf{v}$ are the fluid and particle velocity, respectively (SI unit: m/s). The particle velocity response time τ_p (SI unit: s) is a measure of the time scale for the particle velocity to approach that of the surrounding fluid. For the Stokes drag law,

$$\tau_p = \frac{\rho_p d_p^2}{18\mu}$$

where

- μ is the fluid dynamic viscosity (SI unit: Pa·s),
- ρ_p is the particle density (SI unit: kg/m^3), and
- d_p is the particle diameter (SI unit: m).

The Stokes drag law is applicable when the particles are sufficiently small and move slowly relative to the surrounding fluid. For large, heavy particles with more inertia, an alternative formulation like the Schiller–Naumann drag law might be more appropriate.

GRAVITY FORCE

The **Gravity Force** node also includes the buoyancy force by considering the density of the surrounding fluid ρ (SI unit: kg/m^3). The gravity force, $\mathbf{F}_g$, is thus

$$\mathbf{F}_g = m_p \mathbf{g} \frac{(\rho_p - \rho)}{\rho_p}$$

THERMOPHORETIC FORCE

The **Thermophoretic Force** applies to particles in a nonisothermal flow. The driving mechanism behind this force is the collision of gas molecules on the particle surface. Collisions are more likely to occur on the hotter side of the particle where the average molecular velocity of the gas is greater. This results in a net force toward colder regions of the gas.

In a continuum flow, the thermophoretic force, $\mathbf{F}_{tp}$, is defined as

$$\mathbf{F}_{tp} = -\frac{6\pi d_p \mu^2 C_s \Lambda \nabla T}{\rho(2\Lambda + 1)T}$$

$$\Lambda = \frac{k}{k_p}$$

where

- k (SI unit: W/(m·K)) is the thermal conductivity of the fluid,
- k_p (SI unit: W/(m·K)) is the particle thermal conductivity,
- T (SI unit: K) is the fluid temperature,
- d_p (SI unit: m) is the particle diameter,
- ρ (SI unit: kg/m^3) is the fluid density, and
- C_s is a dimensionless constant equal to 1.17.

The **Thermophoretic Force** node also provides some corrections for high Knudsen number flows, when the mean free path between gas molecule collisions is comparable in size to the particle diameter. However, in the present model, such high Knudsen number corrections can safely be neglected.

Results and Discussion

The gas velocity is plotted in Figure 2 and the temperature in Figure 3. The gas velocity is higher at the outlet than at the inlet due to thermal expansion. The strong heating of the gas by the susceptor causes a decrease in density, so the gas velocity must increase in order for the total mass in the system to be conserved.

The particles begin to move from left to right due to the drag force and also vertically due to the thermophoretic force; see Figure 4. The color of the particle trajectories represents the magnitude of the thermophoretic force. This is because the susceptor temperature is higher than the temperature on the upper surface. The more the temperature of the susceptor increases, the larger the particle free zone which develops just above. The height of the particle-free zone versus the susceptor temperature is plotted in Figure 5 and agrees well with Fig 6a in Ref. 1. The height of the particle-free zone clearly increases as the susceptor temperature increases, indicating an increase in the magnitude of the thermophoretic force.

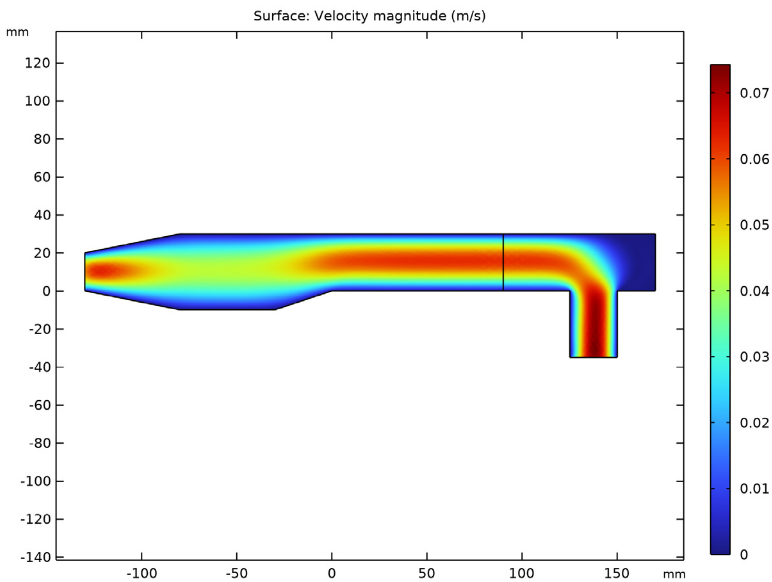

Figure 2: Velocity magnitude.

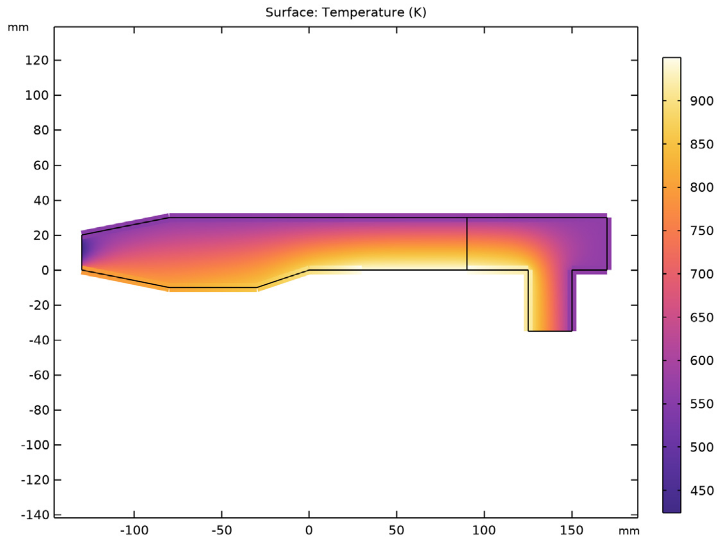

Figure 3: Temperature inside the reactor.

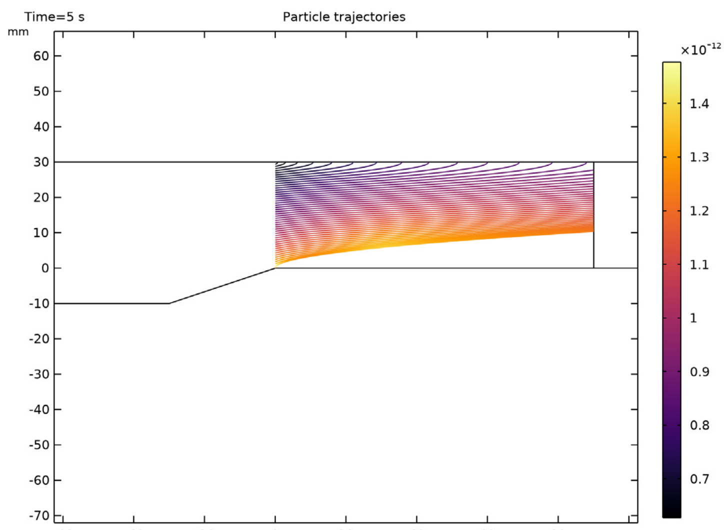

Figure 4: Particle trajectories as they pass over the heated surface.

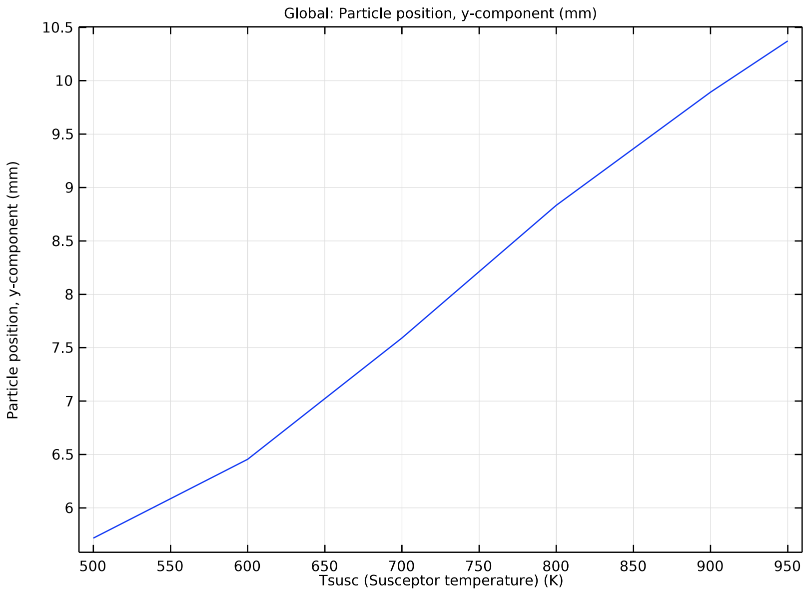

Figure 5: Minimum height of the particles versus temperature.

Notes About the COMSOL Implementation

The model is solved in two stages. First, the gas velocity, pressure, and temperature are computed using a **Stationary** study. Next, the Particle Tracing for Fluid Flow interface is added to the model and the trajectories are computed in a separate **Time Dependent** study. Finally, a **Parametric Sweep** is added with two study steps, a **Stationary** step for the gas flow, followed by a **Time Dependent** step for the particle trajectories. The parametric sweep is performed for a range of susceptor surface temperatures, meaning that both the gas flow and particle trajectories need to be recomputed for each parameter value.

Reference

1. Dimitrios I. Fotiadis, and Klavs F. Jensen, "Thermophoresis of solid particles in horizontal chemical vapor deposition reactors," *J. Crystal Growth*, vol. 102 pp 743–761, 1990.

Application Library path: `Heat_Transfer_Module/Thermal_Processing/thermophoresis`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Fluid Flow** > **Nonisothermal Flow** > **Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `170`.
4 In the **Height** text field, type `30`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 From the **Specify** list, choose **Coordinates**.
4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type 30.

6 Locate the **Endpoint** section. In the **x** text field, type 90.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 50.

4 In the **Height** text field, type 40.

5 Locate the **Position** section. In the **x** text field, type -80.

6 In the **y** text field, type -10.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type -30 30 30 0 0 -10 -10 30.

5 In the **y** text field, type 30 30 30 0 0 -10 -10 30.

6 In the **x** text field, type -30 0 0 0 0 -30 -30 -30.

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type -80 -130 -130 -130 -130 -80 -80 -80.

5 In the **y** text field, type 30 20 20 0 0 -10 -10 30.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type 90.

6 Locate the **Endpoint** section. In the **x** text field, type 90.

7 In the **y** text field, type 30.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `25`.
4 In the **Height** text field, type `35`.
5 Locate the **Position** section. In the **x** text field, type `125`.
6 In the **y** text field, type `-35`.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the objects **pol1**, **pol2**, **r1**, **r2**, and **r3** only.
3 In the **Settings** window for **Union**, locate the **Union** section.
4 Clear the **Keep interior boundaries** checkbox.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.
2 Click the **Zoom Extents** button in the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
Tsusc	950[K]	950 K	Susceptor temperature
d_part	2e-6[m]	2E-6 m	Particle diameter
rho_part	2200[kg/m^3]	2200 kg/m³	Particle density
depth	74[mm]	0.074 m	Model depth

DEFINITIONS

Walls

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.
2 Right-click **Definitions** and choose **Selections** > **Explicit**.
3 In the **Settings** window for **Explicit**, type `Walls` in the **Label** text field.

4. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
5. Select Boundaries 2–9, 12–14, and 16–18 only.

ADD MATERIAL

1. In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Liquids and Gases** > **Gases** > **Hydrogen**.
4. Click the **Add to Component** button in the window toolbar.
5. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER IN FLUIDS (HT)

Temperature 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2. Select Boundary 10 only.
3. In the **Settings** window for **Temperature**, locate the **Temperature** section.
4. In the T_0 text field, type `Tsusc`.

Inflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.
4. In the T_{ustr} text field, type `300`.

Outflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. Select Boundary 15 only.

LAMINAR FLOW (SPF)

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be compressible.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2. In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
3. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Mass flow**.

5 Locate the **Mass Flow** section. From the **Mass flow type** list, choose **Standard flow rate (SCCM)**.

6 In the Q_{sccm} text field, type `2000`.

7 In the M_n text field, type `0.002`.

8 In the d_{bc} text field, type `depth`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 15 only.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `10`.

Thin Layer 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

Since the **Thin Layer** feature has been added, material properties need to be defined on those boundaries.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers** > **Single Layer Material**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic
Density	rho	2700	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	160	W/(m·K)	Basic
Thickness	lth	5[mm]	m	Shell

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

STUDY 1

1 In the **Study** toolbar, click **Compute**.

The default plots which are created automatically are a surface plot of the velocity field (Figure 2), a contour plot of the pressure, a surface temperature plot (Figure 3), and a contour plot of the temperature.

RESULTS

Velocity (spf)

Next, set up the Particle Tracing Interface.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow** > **Particle Tracing** > **Particle Tracing for Fluid Flow (fpt)**.

4 Click the **Add to Component 1** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

PARTICLE TRACING FOR FLUID FLOW (FPT)

Drag Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Drag Force**.

2 In the **Settings** window for **Drag Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Drag Force** section. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

5 From the μ list, choose **Dynamic viscosity (spf)**.

6 Locate the **Model Input** section. From the T list, choose **Temperature (ht)**.

7 From the p_A list, choose **Absolute pressure (spf)**.

Thermophoretic Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Thermophoretic Force**.

2 In the **Settings** window for **Thermophoretic Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Fluid Properties** section. From the μ list, choose **Dynamic viscosity (spf)**.

5 From the ρ list, choose **Density (spf)**.

6 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** checkbox.

7 Locate the **Model Input** section. From the T list, choose **Temperature (ht)**.

8 From the p_A list, choose **Absolute pressure (spf)**.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 11 only.

Gravity Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Gravity Force**.

2 In the **Settings** window for **Gravity Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Gravity Force** section. From the ρ list, choose **Density (spf)**.

Release from Grid 1

1 In the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

3 Click **Y Range**.

4 In the **Range** dialog, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `0.01`.

6 In the **Stop** text field, type `29.99`.

7 In the **Number of values** text field, type `50`.

8 Click **Replace**.

9 In the **Settings** window for **Release from Grid**, locate the **Initial Velocity** section.

10 Specify the $\mathbf{v}_0$ vector as

u	x
v	y

Particle Properties 1

1 In the **Model Builder** window, click **Particle Properties 1**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Properties** section.

3 From the ρ_p list, choose **User defined**. In the associated text field, type `rho_part`.

4 In the d_p text field, type `d_part`.

5 Locate the **Additional Material Properties** section. From the k_p list, choose **User defined**. In the associated text field, type `1.38`.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog, choose **Number of values** from the **Entry method** list.

5 In the **Stop** text field, type `5`.

6 In the **Number of values** text field, type 50.

7 Click **Replace**.

8 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

9 Select the **Modify model configuration for study step** checkbox.

10 In the tree, select **Component 1 (comp1)** > **Laminar Flow (spf)**.

11 Click **Disable in Solvers**.

12 In the tree, select **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)**.

13 Click **Disable in Solvers**.

14 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Nonisothermal Flow 1 (nitf1)**.

15 Click **Disable in Solvers**.

16 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

17 From the **Method** list, choose **Solution**.

18 From the **Study** list, choose **Study 1, Stationary**.

19 In the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories (fpt)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Model Builder** window, expand the **Particle Trajectories (fpt)** node.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Results** > **Particle Trajectories (fpt)** > **Particle Trajectories 1** node, then click **Particle Trajectories 1**.

2 In the **Settings** window for **Particle Trajectories**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

4 Find the **Point style** subsection. From the **Type** list, choose **None**.

5 In the **Particle Trajectories (fpt)** toolbar, click **Plot**.

Color Expression 1

1 In the **Model Builder** window, click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Particle Tracing for Fluid Flow** > **Forces** > **Thermophoretic force - N** > **fpt.thpf1.Ftfy - Thermophoretic force, y-component**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Inferno**.

4 In the **Particle Trajectories (fpt)** toolbar, click **Plot**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Stationary**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 2: Time Dependent

1 In the **Study** toolbar, click **Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Laminar Flow (spf)**.

5 Click **Disable in Solvers**.

6 In the tree, select **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)**.

7 Click **Disable in Solvers**.

8 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Nonisothermal Flow 1 (nitf1)**.

9 Click **Disable in Solvers**.

10 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

11 From the **Method** list, choose **Solution**.

12 From the **Study** list, choose **Study 3, Stationary**.

13 Locate the **Study Settings** section. Click **Range**.

14 In the **Range** dialog, choose **Number of values** from the **Entry method** list.

15 In the **Stop** text field, type 5.

16 In the **Number of values** text field, type 10.

17 Click **Replace**.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Tsusc (Susceptor temperature)	500 600 700 800 900 950	K

5 In the **Study** toolbar, click **Compute**.

RESULTS

Particle 3

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** > **Particle 2** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 11 only.

Minimum 1

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation** > **Minimum**.

2 In the **Settings** window for **Minimum**, locate the **Data** section.

3 From the **Dataset** list, choose **Particle 3**.

4 Locate the **Settings** section. From the **Geometry level** list, choose **Point**.

Particle Position

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Particle Position` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Minimum 1**.

4 From the **Time selection** list, choose **Last**.

Global 1

1 Right-click **Particle Position** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Particle Tracing for Fluid Flow** > **Particle position** > **qy - Particle position, y-component - m**.

3 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

4 In the **Particle Position** toolbar, click  **Plot**.

5 Click to expand the **Legends** section. Clear the **Show legends** checkbox.